CFD Letters www.cfdl.issres.net Vol. 2(4) – December 2010

A CFD Validation of Fire Dynamics Simulator for Corner Fire

Pavan K. Sharma[C], B. Gera, R. K. Singh
Reactor Safety Division, Engg. Hall-7, Bhabha Atomic Research Centre Trombay, Mumbai

Received: 06/07/2010 – Revised 25/08/2010 – Accepted 13/10/2010

Abstract

A computational study has been carried out for predicting the behaviour of a corner fire source for a reported experiment using a field model based code Fire Dynamics Simulator (FDS). Time dependent temperature is predicted along with the resulting changes in the plume structure. The flux falling on the wall was also observed. The analysis has been carried out with the correct value of the grid size based on earlier experiences and also by performing a grid sensitivity study. The predicted temperatures of the two scenarios at two points by the current analysis are in very good agreement with the earlier reported experimental data and numerical prediction. The studies have extended the utility of field model based tools to model the particular separate effect phenomenon like corner for one such situation and validate against experimental data. The present study have several applications in such as room fires, hydrogen transport in nuclear reactor containment, natural convection in building flows etc. The present approach uses the advanced Large Eddy Simulation (LES) based CFD turbulence model. The paper presents brief description of the code FDS, details of the computational model along with the discussions on the results obtained under these studies. The validated CFD based procedure has been used for solving various problems enclosure fire, ventilated fire and open fire from nuclear industry which are however not included in the present paper.

Keywords: CFD; Fire dynamics simulator; corner fire; large eddy simulation.

1. Introduction

The fundamental knowledge of fluid flow and heat transfer associated with fire growth, and the related numerical modeling is vital in the assessment of fire safety. The behaviour of hot combustion products which rise up in the form of buoyant plume is one such, among several other important aspects. Research studies on free, confined and non confined corner fires have been of great importance due to their high probability of occurrence in engineering industry and even in the domestic places. Some of such situations are accidental release of flammable gases in industrial fires and fires in enclosures. Fig. 1 shows the schematic of the development of corner fire. The nature of the plume above the pool fire is different in free plume and enclosure plume development. In enclosure plume development the plume behaviour is further function of location of the pool fire in the room i.e. corner, wall, centre etc[1]. In case of corner fire the air entrainment is smaller than

[C] Corresponding Author: Pavan Sharma
Email: pksharma@barc.gov.in
 PII: S2180-1363(10)24137-X

that in free plume condition because of wall restrictions [2]. Authors themselves have studied the buoyant plume rise time for confined and unconfined plume in centre, wall and corner situation [3]. Apart from the earlier experimental work the advance CFD techniques are now being capable of studying this phenomenon in details.

Several special-purpose and general-purpose software packages have been developed in recent years. Fire Dynamics Simulator (FDS) [4] and Smart Fire [5] are popular special-purpose software, and CFX [6], FLUENT [7] and PHOENICS [8] are among the popular general-purpose CFD software. This study took one of the special-purpose CFD software packages, FDS [4], as an example, to investigate the feasibility of using this program for the modelling of a corner fire in an enclosure. RANS CFD packages [9] [10] [1] [12] however, are quite commonly used in field modelling of fire phenomena.

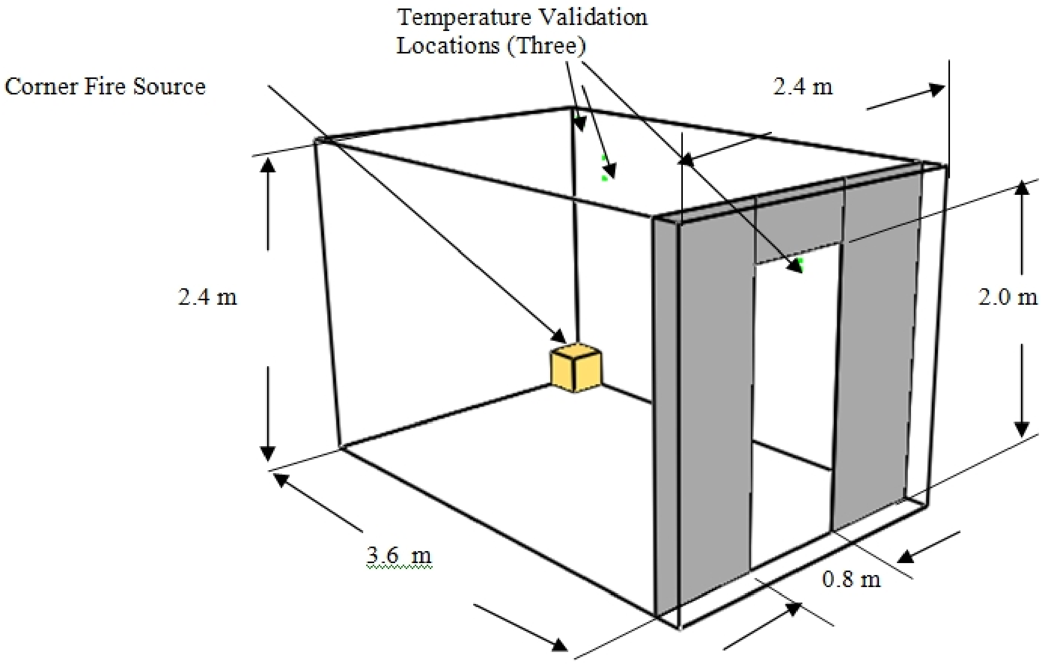


Figure 1. Schematic diagram of corner fire development

The main criticism leveled at the RANS approach targets the validity of the turbulence models employed to provide closure to the governing equation set as these turbulence models contain empirically determined parameters that can only be considered applicable for the specific flow cases where they have been validated and this aspects makes the range of validity for the RANS CFD approach application specific [13][14][15] . Large Eddy Simulation (LES) is a branch of Computational Fluid Dynamics (CFD). It differs from other CFD approaches, in that LES explicitly calculates the large-scale turbulent flow structures from first principles. The small-scale turbulent motion not calculated directly from the governing equations has its influence on the resolved flow field modeled. The simplest and most common form of SGS turbulence model used in LES is the Smagorinsky model [16] [17]. This is the default SGS model used in the FDS fire code. The Smagorinsky model uses the eddy viscosity approach to quantify the stresses that the turbulent velocity fluctuations place on the mean flow. In this modelling approach, the Reynolds stresses are assumed to relate to the local mean velocity gradient. The present work has been carried out using the code FDS which has been validated for modelling buoyant plumes against the experimental data by Baum and Macferrey mentioned in technical reference guide [4]. The code FDS have been extensively validated using salt water modeling experiments[18,19], gravity current modeling [20,21,22], isolated plumes[23,24,25,26], buoyant plume in cross flow [27], enclosure fire dynamics [28,29,30] and by qualitative observations in above mentioned studies. A detailed systematic validation exercise for the FDS hydrodynamic model can be found in reference [31].

Authors have also used the validated tool for various applications in nuclear industry [32].

The aim of present paper is to study the corner fire characteristics in a compartment which have been an area of keen interest worldwide. Numerical prediction of above situations has been compared with the experimental and numerical results available from open literature. Simulations have been carried with correct grid sizes based on our earlier studies [3].

2. Fire Dynamics Simulator (FDS)

Fire Dynamics Simulator (FDS) is a field model based software developed by NIST (USA) and has built-in dedicated computational fluid dynamics (CFD) model to describe and analyse fire in multiple compartment geometries. The code solves numerically the set of governing equations of mass, momentum (Navier-Stroke) and energy conservation appropriate for low-speed, thermally-driven flows of multispecies gas mixture to describe the smoke and heat transport arising from fires. The details of formulation of the equations and the choice of numerical algorithm available are contained in a companion document, called Fire Dynamics Simulator – User and Mathematical Reference Guide [4]. Fire Dynamics Simulator systematic validation and verification are described elsewhere [33, 34].

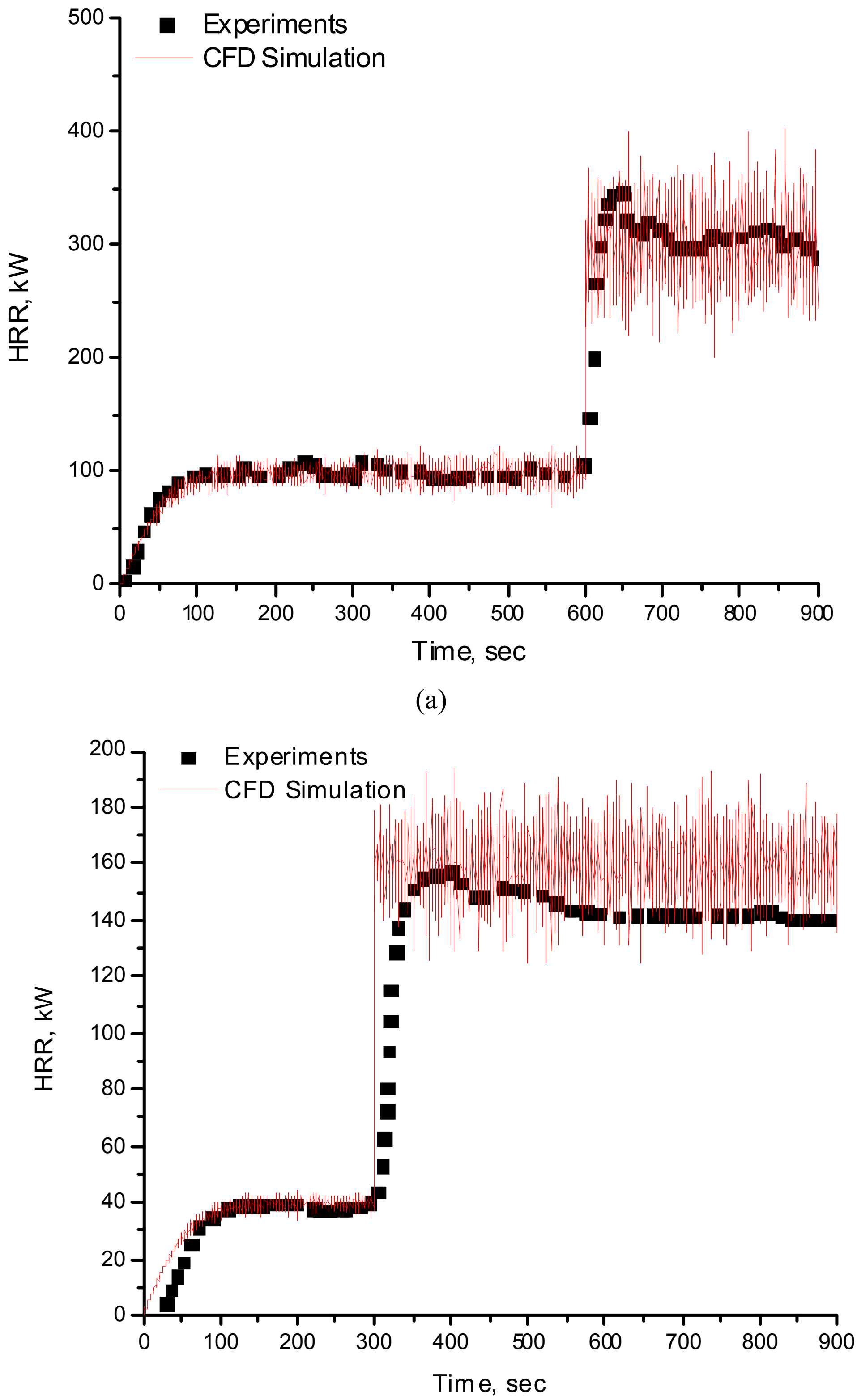
3. Validation Case Study for Analysis

This investigation is based on two fire experimental tests that were conducted by the CSIRO fire research group [35]. The test room was 3.6m long x 2.4m wide x 2.4m high with a 0.8m wide x 2.0m high door. Figure 1 gives a modeled view of the ISO test room [35]. In this study, two experiments with plasterboard wall lining materials are considered, where there was no fire spread and the heat release was contributed only by a methane burner in a corner of the room. The burner was located in the corner opposite the door opening, and the burner dimensions were 0.3mx0.3mx0.3m. Two burner heat release rate profiles were used in the experiments. In the ISO test method [35], named as case A hereafter, the supplied methane generated a heat release rate (HRR) of 100kW in the first ten minutes, which was then increased to 300kW and maintained at this rate during the following ten minutes. In the ASTM method [35], named as case B hereafter, the supplied methane generated a heat release rate of 40kW during the first five minutes, which was then increased to 160kW, and was maintained at this level for 10 minutes. The gas temperature development history at several locations below the ceiling was recorded with type K thermocouples at 5-second intervals. These monitor points were located 0.05m below the ceiling centre, 0.1m below the top of the doorway centre, and 0.05m below the ceiling directly above the burner. The recorded time-dependent temperature data formed the basis for the validation of FDS software package. The simulated heat release from experiments and heat release rate which actually has gone in for modelling is shown in Figure 2 a and Figure 2 b.

4. Computational Details

FDS was used as a tool to calculate the temperature field generated by the burner inside the room. The model used in this study does not incorporate fire spread, and the experiments also did not result in flame spread on the linings, which were non-combustible.

Wall was modeled as heat conducting media using conjugate heat transfer approach. For this, the computational domain was made up of the indoor gas domain, 0.1m-thick ceiling, 0.1m-thick walls and 0.1m-deep solid floor. In this case, the computational domain and the boundary were extended to the exterior wall surface to take into account the heat transfer into the wall. To eliminate the influence of the boundary conditions imposed on the doorway plume region, the computation domain was extended few grids (cm) beyond the door, where pressure boundary conditions were applied.



(a)

(b)

Figure 2. (a) Case A: Heat Release Rate From Experiment[35,36] and simulated values and (b) Case B: Heat Release Rate From Experiment[35,36] and simulated values

FDS solves the radiation Transport equation (RTE) for radiation modeling. As the fire-generated buoyancy driven flow is turbulent, and results into natural convection, the LES turbulence model was employed to resolve the subscale turbulence.

The fire source is taken as the input parameter, which is a stable heat release rate that was designed to represent the experimental measured HRR by the calorimeter. As a fire scenario is transient, a smaller time step is taken in the initial stage of the fire when the temperature and flow field development is fast, and a larger time step is taken for the steady developing stage.

A Grid sensitivity study was carried out. To start with first simulation was carried out with fine grids of 0.1 m based on our earlier experience. The measured temperatures with this grid resolution were reasonably close to the experimental data. Still fined grids were used with 0.05m being a validation study. The Fig. 3 depicts the results of the grid sensitivity study. Subsequently in both the validations computational domain was divided in 76X48X48 cells to have a grid resolution of 0.05 m. The penalty for doing this was the high computational time. Each case study took about 40 -45 hours on Intel Xeon 2.5 Ghz Twin processor window based hardware which used 8 threads with openMP version of Fire Dynamics Simulator. A fixed time step of 0.5 sec was used in both of the simulation. However the adoptive time step feature was also used. In case if the based on the internal modified CFL type of criteria[4] if a small time step then 0.5 is was required the code automatically uses the required time step. The temperature and the gas flow development history at different locations below the ceiling was recorded in the results of the CFD modelling. Another objective of having 0.05 m grid was to resolve the grid near the wall as one of the reported monitoring location was 0.05 m away from the ceiling. The CFD modeling accuracy is evaluated on the basis of the predicted gas temperature development history at these locations.

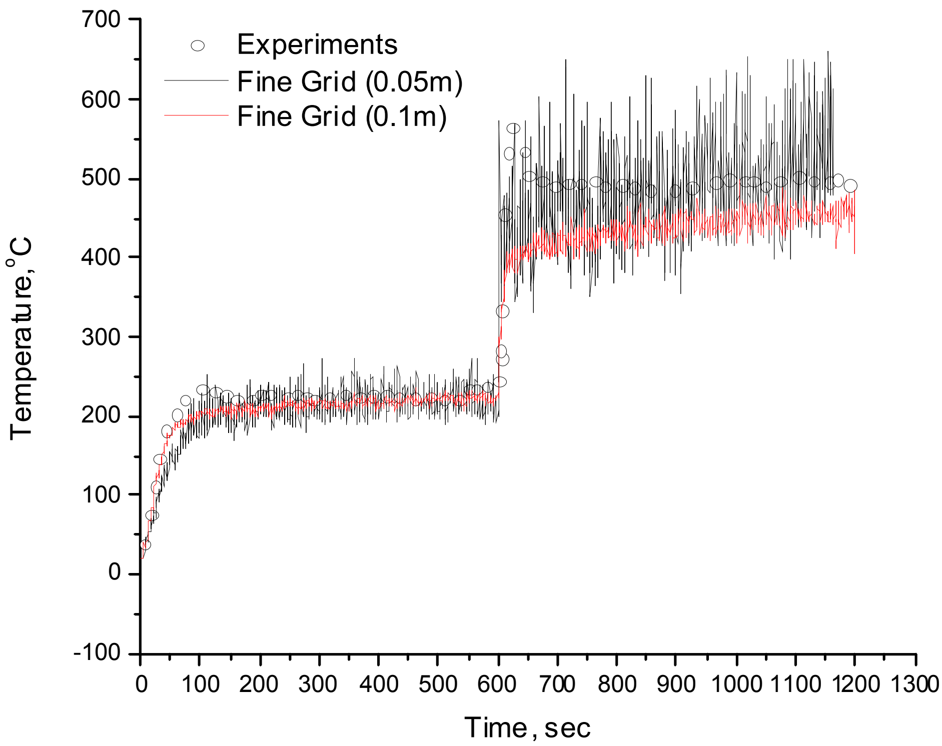


Figure: 3 Gas temperature at 0.05m below the ceiling centre for case A for 0.1m and 0.05 m grid fineness (Grid sensitivity)

5. Results and Discussion for Validation Study

Based on the CSIRO experiment [35], the temperature development histories at several key locations below the ceiling were taken as the criteria for comparison. As the flow field and the temperature field interact with each other, the accuracy resolution of the transient temperature field can serve as an indicator to evaluate the applicability of the CFD software package.

All the CFD predicted curves presented in Figure 4, 5 and 6 for case A. Figure 4 depicts the FDS predicted air temperature and its comparison with reported numerical and experimental results [35] at a location 0.05m below the ceiling centre. Temperature history at this location shows that gas temperature increased very fast in first few seconds after the ignition. After that, when the heat release from the burner is stable, little temperature increase was detected. This is because the heat released from the burner is dissipated through natural ventilation and radiative heat transfer into the wall surfaces, so the gas temperature in the room achieved a balance. The initial growth in first 100 seconds in FDS prediction is assumed to be happening by t^2 profile similar to experiment fire growth takes some time and follows. However in second step change the t^2 profile has not been assumed as authors assumes due to high temperature around burners the lead time of combustion may not be significant. According to Figure 5, during the stable burning period, the gas temperature development had been correctly predicted by FDS. Gas temperature above the burner and below the ceiling for case A also in good agreement with the experimental data (Figure 6). Figure 6 presents the comparison of the CFD predicted and the measured gas temperature development history at a location 0.1m below the top of the doorway centre. This figure also shows the fast gas temperature development during the first 100 seconds after ignition; the gas temperature remained stable with little temperature change during the following ten minutes, these results agrees well with the experiment.

All the CFD results presented in Figure 7, 8 and 9 are for case B with the conjugate heat transfer boundary conditions, and the fine mesh was used. Figure 7 gives a comparison of the FDS prediction and reported experimental and numerical temperatures development history at a location 0.05m below the ceiling centre. During the early phase and later phase FDS predicted gas temperature agrees well with the experimental values. Similar observation can be drawn from figure 8 for gas temperature above the burner and below the ceiling for case B.

Figure 9 presents the CFD predicted and the measured gas temperature at a location 0.1m below the top of the doorway centre. The results agree well with the experiments for there utilization in engineering accuracy in all the cases in spite of the uncertainties in of the heat transfer computation, fuel combustion t^2 profile growth assumption in a assumed time duration for burner and as well as the experimental measurement accuracy. As thermal radiation, heat conduction and convective heat transfer co-exist in these cases, all the sub-models needs to be validated separately for the modelling of the thermal radiation, convective heat transfer and heat conduction, which is the research topic for further investigation.

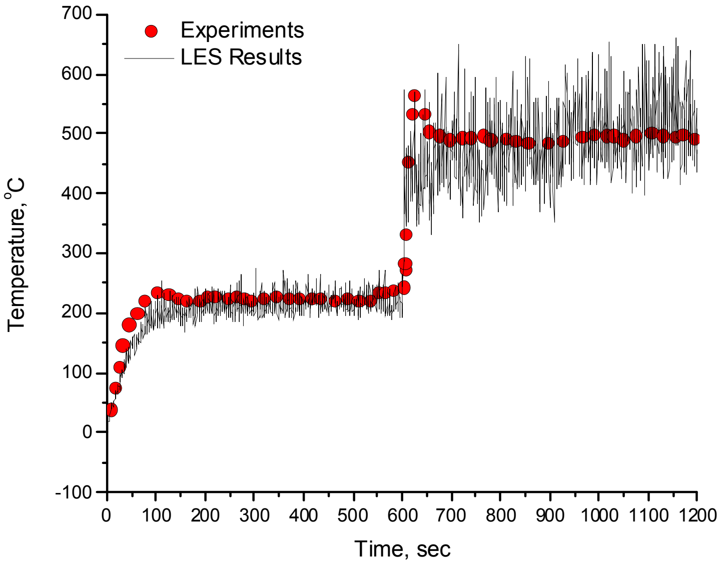


Figure : 4 Gas temperature at 0.05m below the ceiling centre for case A

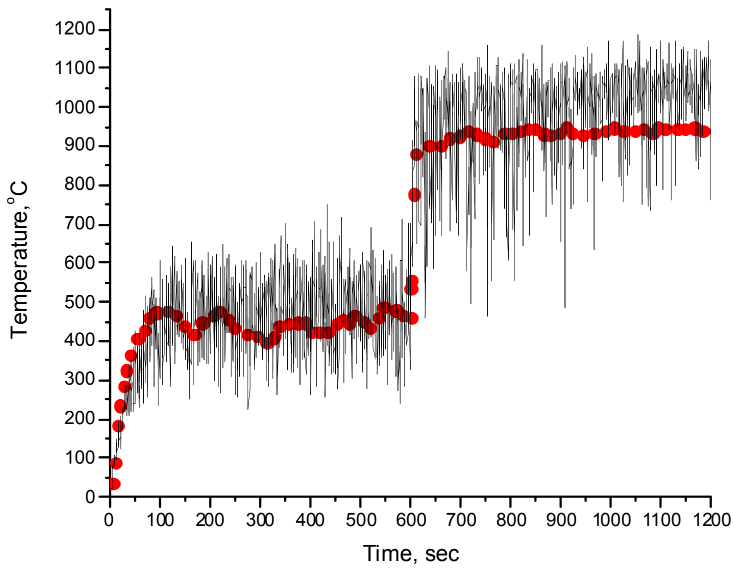


Figure : 5 Gas temperature above the burner and 0.05m below the ceiling for case A

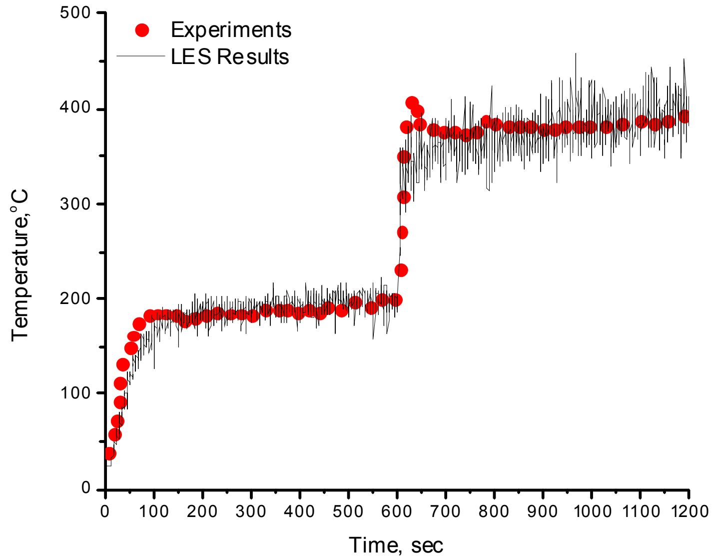


Figure : 6 Gas temperature at 0.1m below the top of the door centre for case A

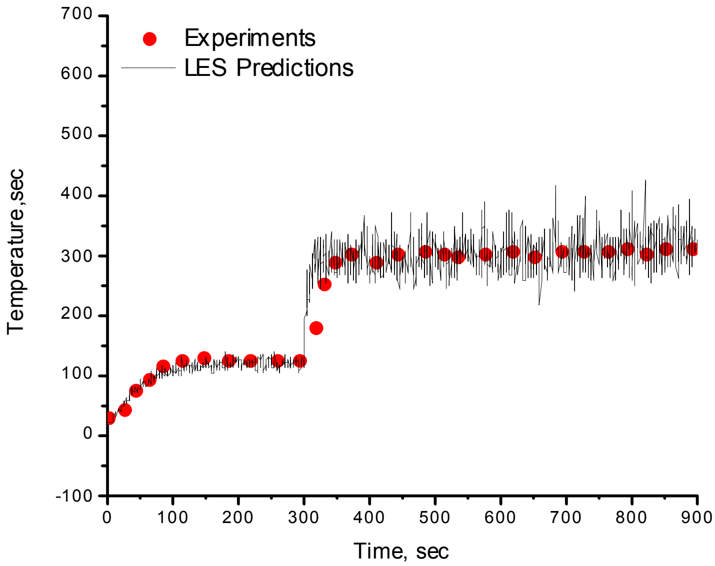


Figure : 7 Gas temperature at 0.05m below the ceiling centre for case B

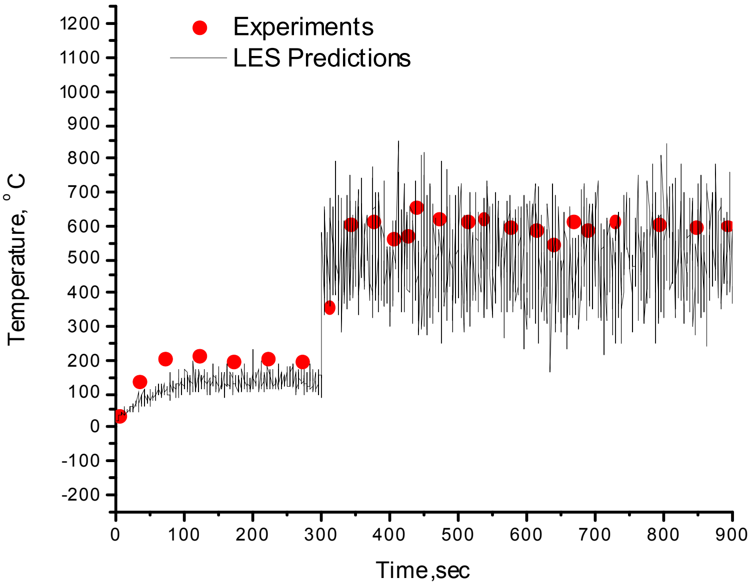


Figure : 8 Gas temperature above the burner and 0.05m below the ceiling for case B

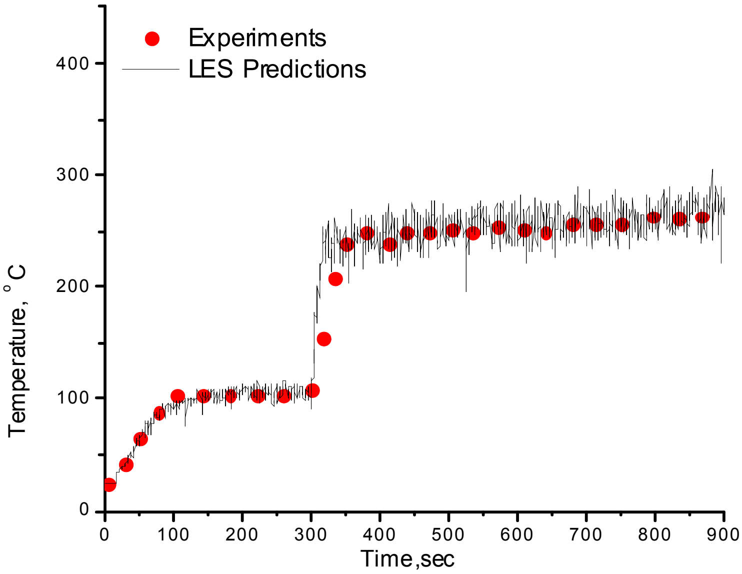


Figure : 9 Gas temperature at 0.1m below the top of the door centre for case B

Figures 10-13 show the instantaneous temperature contours at a vertical plane passing through the fire source location at different time for case A. Figure 14-17 show the instantaneous temperature contours at a vertical mid plane passing at a vertical mid plane passing through the centre of the room at different time for case A. These figures show the temperature variation at the opening location. From these counters it is clear that the room is divided in two zones one is heated zone in the upper half of the room other is relatively cold zone in the lower half of the room. Bi-directional flow through the large vertical opening is clearly manifest.

From the above discussion, it can be concluded that FDS is an excellent tool for prediction of gas temperature for such scenarios. As fire scenario is always related to high temperature and strong thermal radiation, both measurement and CFD modeling can have uncertainties. To investigate and validate a fire model, more comprehensive research work experimentally as well as numerically is necessary.

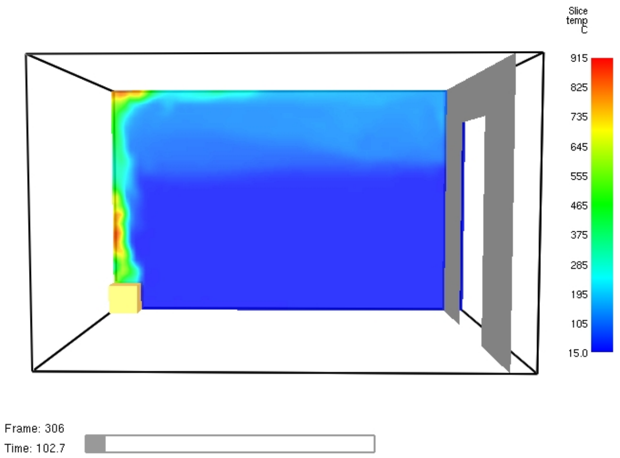


Figure 10 Temperature contours at a vertical plane passing through the fire source location at 102.7 sec for case A

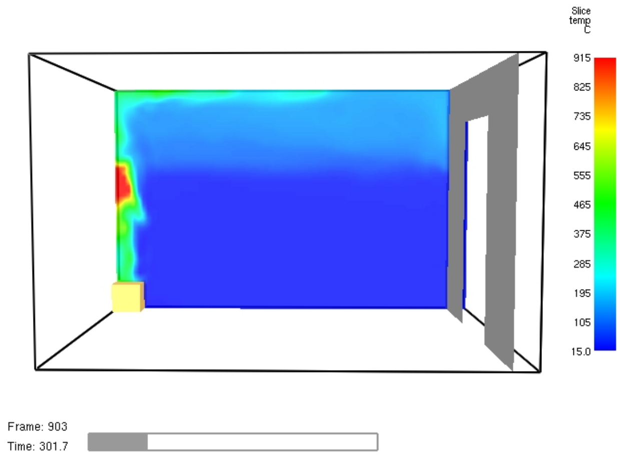


Figure 11 Temperature contours at a vertical plane passing through the fire source location at 301.7 sec for case A

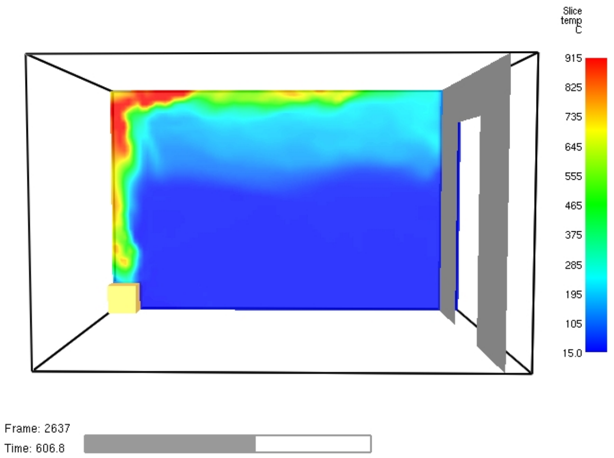


Figure 12 Temperature contours at a vertical plane passing through the fire source location at 606.8 sec for case A

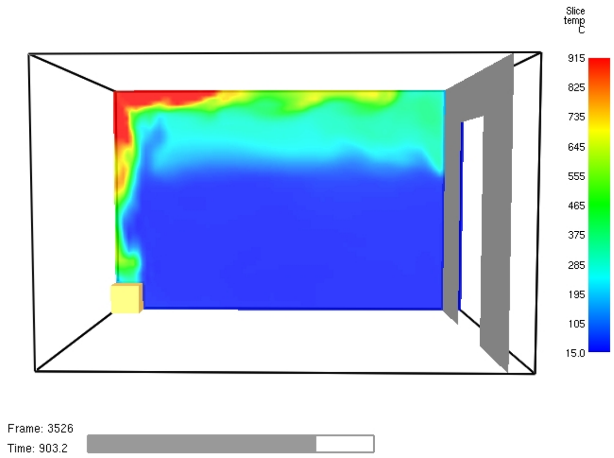


Figure 13 Temperature contours at a vertical plane passing through the fire source location at 903.2 sec for case A

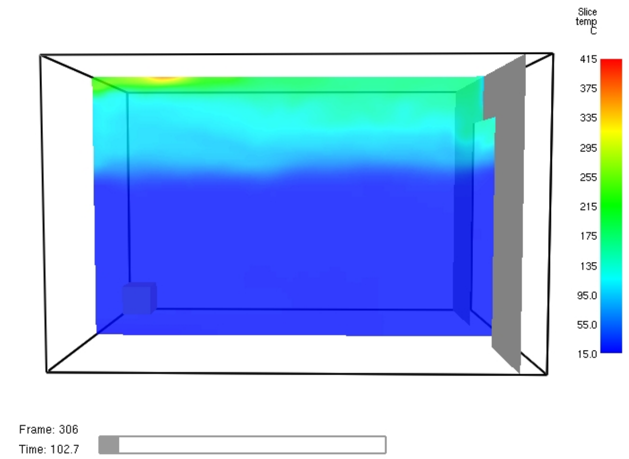


Figure 14 Temperature contours at a vertical mid plane passing through the centre of the room at 102.7 sec for case A

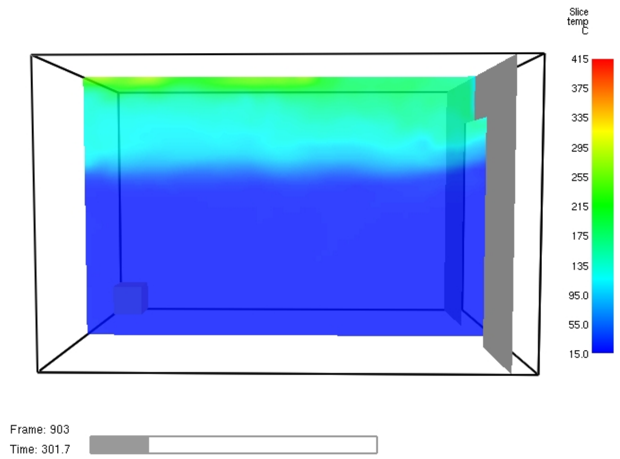


Figure 15 Temperature contours at a vertical mid plane passing through the centre of the room at 301.7 sec for case A

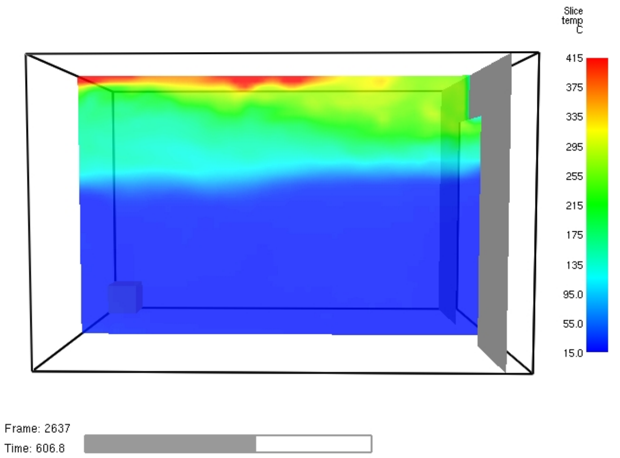


Figure 16 Temperature contours at a vertical mid plane passing through the centre of the room at 606.8 sec for case A

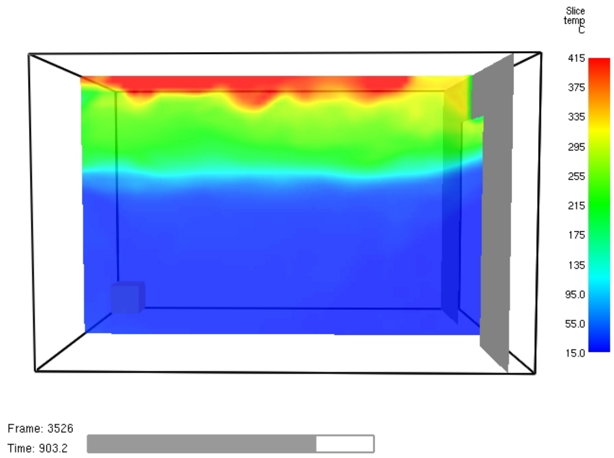


Figure 17 Temperature contours at a vertical mid plane passing through the centre of the room at 903.2 sec for case A

6. Conclusion

In the present study one of the important issues of fire modeling was discussed. From the above discussion, following conclusions can be drawn. Reasonable temperature field can be obtained for the modelling of a fire in a test room using the FDS software package. The solid wall is included into the computation domain as the heat conduction into the wall accounted for a large portion of the total heat transfer, and this has improved the CFD modeling accuracy of the indoor gas temperature development as the reported adiabatic boundary condition at the wall predicts the higher temperature[35]. The LES turbulence model is suitable for the modelling of buoyancy-generated turbulence, if the meshing size is sufficient to resolve the subscale turbulence.

References

[1] Hasemi, Y.; Tokunaga, T., Some Experimental Aspects of Turbulent Diffusion Flames and Buoyant Plumes from Fire Sources Against a Wall and in a Corner of Walls, Combustion Science and Technology, vol(40)1, 1984, p. 1 -18.

[2] Sugawa, O.; Tobari M., Behaviour of Flame/Plume Flow in and near Corner Fire-Entrainment Coefficient for Corner Fire, UJNR Meeting, NISTIR 6588, 2000.

[3] Sharma, Pavan K.; Ghosh, A. K.; Kushwaha, H. S., A Computational Fluid Dynamics Study of Buoyant Plume and its Rise Time, 32nd National Conference on Fluid Mechanics & Fluid Power, College of Engineering , Osmanabad, Maharashtra, 2005.

[4] McGrattan Kevin; McDermott Randall; Hostikka Simo; Floyd Jason, Fire Dynamics Simulator (Version 5) , User's Guide NIST Special Publication 1019-5, In cooperation with: VTT Technical Research Centre of Finland National Institute of Standards and Technology, Gaithersburg, Maryland, June 23, 2010.

[5] Ewer J ; Galea E R, An Intelligent CFD Based Fire Model, Journal of Fire Protection Engineering, vol(1), 1999, p.12-27.

[6] Simcox S; Wilkes et al, Computer Simulation of the Flows of Hot Gases from the Fires at King's Cross Underground Station, Fire Safety Journal, vol(18), 1992, p.49-73.

[7] Barrero D.; Ozell B.; and Reggio M. , On CFD and Graphic Animation for Fire Simulation, The Eleventh Annual Conference of the CFD Society of Canada, Vancouver, BC, 2003.

[8] Glynn D.R.; Eckford D.C. and Pope C.W., Smoke Concentrations and Air Temperatures Generated by a Fire on a Train in a Tunnel, The PHOENICS Journal of Computational Fluid Dynamics and its Applications, vol(9) 1, 1996, p. 157-168.

[9] Kerrison, L.; MaWhinney, N.; Galea, E.R.; Hoffmann, N.; and Patel, M.K., A Comparison of a FLOW3D Based Fire Field Model with Experimental Compartment Fire Data, Fire Safety Journal, vol(23), 1994, p. 387-411.

[10] Kumar, S., Hoffmann, N. and Cox, G., "Some Validation of Jasmine for Fires in Hospital Wards", Numerical Solution of Fluid Flow and Heat/Mass Transfer Processes, Springer-Verlag, Berlin, p.159, 1985.

[11] Welch, S.; and Rubini, P.A., SOFIE 2.1 User's Manual, Cranfield University, Cranfield, United Kingdom, 1996.

[12] Galea, E.R.; Knight, B.; Patel, M.K.; Ewer, J.; Pitridis, M.; and Taylor, S., SMARTFIRE v2.0 User Guide and Technical Manual, Fire Safety Engineering Group, University of Greenwich, UK, 1998.

[13] Webby A. T. ; Mansourz N. N., Towards LES Models of Jets and Plumes, Center for Turbulence Research, Annual Research Briefs 2000.

[14] Kenjereˇs S ; Kemal Hanjali´c K., Tackling Complex Turbulent Flows with Transient RANS, Fluid Dynamics Research, vol (41) 2009, p.32.

[15] Alfonsi G, Reynolds-Averaged Navier--Stokes Equations for Turbulence Modeling, Appl. Mech. Vol(62) 2009, p. 040802.

[16] Ferziger, J.H., Subgrid-Scale Modeling' In: Large Eddy Simulation of Complex Engineering and Geophysical Flows". Editors: B. Galperin and S.A. Orszag, Cambridge University Press, Cambridge, 1993, p. 37-54.

[17] Smagorinsky, J., General Circulation Experiments with the Primitive Equations, Monthly Weather Review, vol(91), 99-165.

[18] Baum, H.R.; and Rehm, R.G., Numerical Computation of Large-Scale Fire Induced Flows, In: Eighth International Conference on Numerical Methods in Fluid Dynamics, Editors: E. Krause, Lecture Notes in Physics vol(170), Springer-Verlag, Berlin, 1982,p. 124-130.

[19] Baum, H.R.; Rehm, R.G.; and Mulholland, G.W., Computation of Fire Induced Flow and Smoke Coagulation", Nineteenth Symposium (International) on Combustion/The Combustion Institute, 1982, pp 921-931.2–22.

[20] Rehm, R.G.; Tang, H.C.; Baum, H.R.; Sims, J.S.; and Corley D.M., A Boussinesq Algorithm for Enclosed Buoyant Convection in Two Dimensions", NISTIR 4540, U.S. Department of Commerce, National Institute of Standards and Technology, Computing and Applied Mathematics Laboratory, Gaithersburg, MD 20899, U.S.A., 1991.

[21] Baum, H.R.; McGrattan, K.B.; and Rehm, R.G., Mathematical Modelling and Computer Simulation of Fire Phenomena, Fire Safety Science – Proceedings of the Fourth International Symposium, Ottawa, Ontario, Canada, 1994, p. 185-193.

[22] Rehm, R.G.; McGrattan, K.B.; Baum, H.R.; and Cassel K.W., Transport by Gravity Currents in Building Fires, Fire Safety Science – Proceedings of the Fifth International Symposium, 1997, p. 391-402.

[23] Baum, H.R.; McGrattan, K.B.; and Rehm, R.G., Large Eddy Simulations of Smoke Movement in Three Dimensions, Conference Proceedings of the Seventh International Interflam Conference, Interscience Communications, London, 1996, p. 189-198.

[24] Baum, H.R.; McGrattan, K.B.; and Rehm, R.G., Three Dimensional Simulations of Fire Plume Dynamics, Fire Safety Science – Proceedings of the Fifth International Symposium, 1997, p. 511-522.

[25] Baum, H.R., Large Eddy Simulations of Fires – from Concepts to Computations, Fire Protection Engineering, vol(6), 1999, p. 36-42.

[26] McGrattan, K.B.; Baum, H.R.; and Rehm, R.G., Large Eddy Simulations of Smoke Movement, Fire Safety Journal, vol(30), 1998, p.161-178.

[27] Sharma, Pavan K.; Markandeya S. G.; Ghosh, A. K.; Kushwaha, H. S.; and Venkat Raj V., A Computational Fluid Dynamics Study of a Buoyant Plume in a Cross Flow Condition, 28th

National Conference on Fluid Mechanics and Fluid Power, Punjab University, Chandigarh, 2001.

[28] Wang Lei; Pu Jin-yun, Computing on Fire-induced Smoke Flow in Irregular Confined Space, International Forum on Computer Science-Technology and Applications, 2009.

[29] Baum, H.R.; McGrattan, K.B.; and Rehm, R.G., Three Dimensional Simulations of Fire Plume Dynamics, Fire Safety Science – Proceedings of the Fifth International Symposium, 1997, p. 511-522.

[30] Spearpoint, M.; Mowrer, F. W.; and McGrattan K, Simulation of a Compartment Flashover Fire Using Hand Calculations, Zone Models and a Field Model, International Conference on Fire Research and Engineering (ICFRE3), Third (3rd). Proceedings. Society of Fire Protection Engineers (SFPE), National Institute of Standard and Technology (NIST) and International Association of Fire Safety Science (IAFSS), 1999, Chicago, IL, Society of Fire Protection Engineers, Boston, MA.

[31] Clement, J. M., Experimental Verification of the Fire Dynamics Simulator (FDS) Hydrodynamic Model, Ph D. Thesis, University of Canterbury, Christchurch, New Zealand, 2000.

[32] Sharma, Pavan K.; Ghosh, A. K.; Singh R. K.;, Kushwaha, H. S., A CFD Validation Study of Pool Fire and Few Applications In Nuclear Industry, Discussion meet on "Fire Modeling", AERB Mumbai., 2007.

[33] McGrattan Kevin; Hostikka Simo; Floyd Jason; Baum Howard; and Rehm Ronald, Fire Dynamics Simulator (Version 5) Volume -2, Validation, Technical Reference Guide NIST Special Publication 1018-5, In cooperation with: VTT Technical Research Centre of Finland National Institute of Standards and Technology, Gaithersburg, Maryland, 2007.

[34] McGrattan Kevin; Hostikka Simo; Floyd Jason; Baum Howard; and Rehm Ronald, Fire Dynamics Simulator (Version 5) Volume -3, Verification, Technical Reference Guide NIST Special Publication 1018-5, In cooperation with: VTT Technical Research Centre of Finland National Institute of Standards and Technology, Gaithersburg, Maryland, 2007.

[35] Webb, A.K.; Dowling, V.P.; and McArthur, N. A., Fire Performances of Materials, internal CSIRO report by Fire Science & Technology, North Ryde, CSIRO, 1999.

[36] Liu, Yunlong; and Apte Vivek, Evaluation of PHOENICS CFD Fire Model against Room Corner Fire Experiments, Proceedings of the International PHOENICS user conference, Melbourne, Australia, 2004.